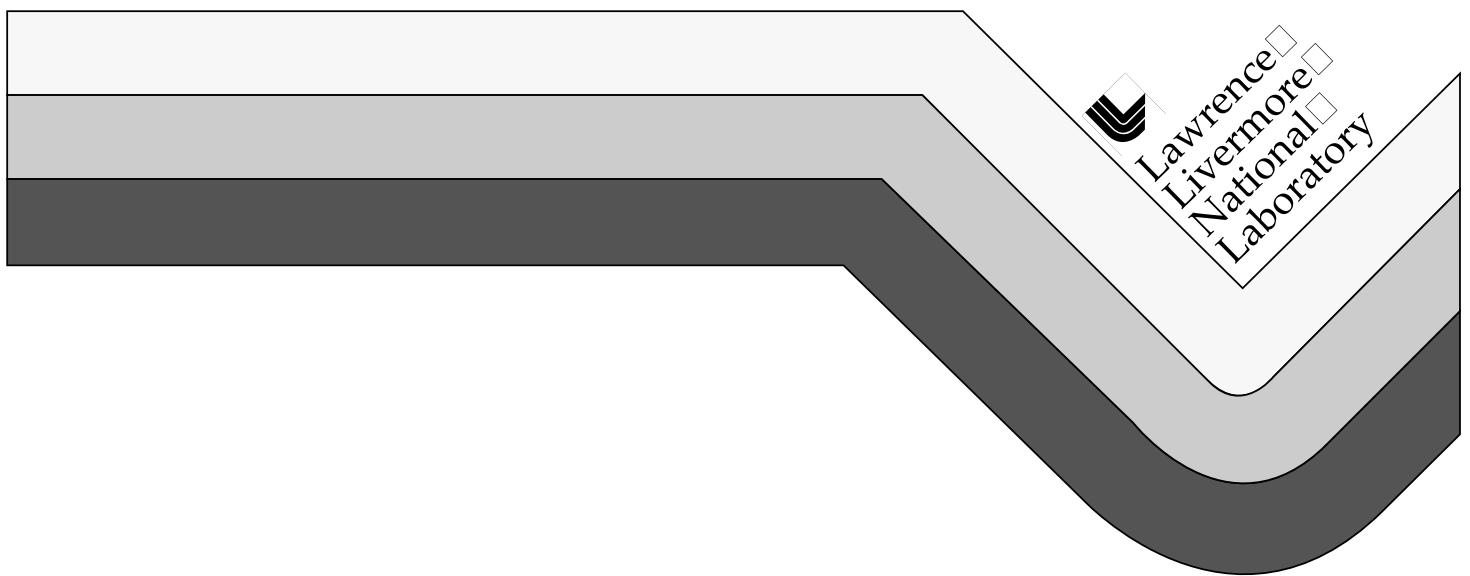


Reference Manual for the NUFT Flow and Transport Code, Version 2.0

John J. Nitao

April 1998



DISCLAIMER

This document was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor the University of California nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or the University of California, and shall not be used for advertising or product endorsement purposes.

**Reference Manual for the
NUFT Flow and Transport Code, Version 2.0**

**John J. Nitaو
Earth and Environmental Sciences
Lawrence Livermore National Laboratory**

April 1998

YMP9804051

Work performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract W-7405-Eng-48. This work is supported by Yucca Mountain Site Characterization Project, LLNL.

Contents

Acknowledgments	ii
1. Introduction.....	1
2. The Syntax of the Input Data.....	3
3. How to Read the Input Documentation	8
4. Basic Elements of the Input File.....	11
5. Input Data Documentation.....	13
5.1 Mesh-Generation Parameters.....	14
5.2 Time-Stepping Parameters.....	22
5.3 Parameters for Numerical Methods	24
5.4 Output Specifications.....	32
5.5 Specifying Initial Conditions.....	39
5.6 Setting Rock Properties	41
5.7 Setting Source Terms.....	42
5.8 Specifying Boundary Conditions.....	45
5.9 Other Options.....	48
6. Running Flow and Transport Sequentially.....	49
7. References.....	50
Appendix A Format of the Mesh File.	56
Appendix B Numerical Algorithms Used.....	58

Acknowledgments

The author wishes to thank the following organizations for supporting the documentation and verification of the NUFT code: Waterways Experimental Station of the U.S. Army Corps of Engineers, the Environmental Restoration Division at the Lawrence Livermore National Laboratory (LLNL), the DOD/DOE Strategic Environmental Research and Development Program (SERDP), and the DOE Yucca Mountain Project. The initial concepts of the code were developed under the LLNL Institutional Research and Development program.

Preparation of this manual was made possible by the administrative assistance of Adrienne F. Ridolfi and the editorial assistance of Karen L. Lew.

1. Introduction

NUFT (Nonisothermal Unsaturated-Saturated Flow and Transport model) is a suite of multi-phase, multicomponent models for numerical solution of non-isothermal flow and transport in porous media with application to subsurface contaminant-transport problems. These distinct models are imbedded in a single code to utilize a common set of utility routines and input file format.

Currently, the code runs on the Unix and DOS operating systems. Versions have been successfully compiled and tested for IBM-PC compatibles, Cray Unicos, and the following workstations: Sun, Hewlett-Packard, IBM Risc/6000, Silicon Graphics, DEC Alpha. Each set of related models is called a module and has its own user's manual that documents any particular features and input data specific to that module. This reference manual for NUFT documents the general numerical algorithms used and gives the documentation of the input to the model common to all or most modules, including options not described in the user's manual for each module.

The following modules are available:

- UCSAT—unconfined and confined saturated flow model
- US1P—single-phase unsaturated flow (Richard's equation)
- US1C—single-component contaminant transport
- USNT—NP-phase, NC-component with thermal option

It is recommended that new users of NUFT read the user's manual (*User's Manual for US1 Module of the NUFT Code, Version 2.0*, and *User's Manual for USNT Module of the NUFT Code, Version 2.0*) before reading this reference manual.

An integrated, finite-difference, spatial discretization is used to solve the balance equations. The resulting nonlinear equation is solved at each time by the Newton-Raphson method. Options for solution of the linear equations at each iteration are direct-banded solution and preconditioned-conjugate gradient method with various preconditioning schemes.

The model can solve one-, two-, or three-dimensional problems. Future plans include incorporation of capillary hysteresis, nonorthogonal mesh discretization, finite elements, and nonlinear solid sorption isotherms.

The first stage of code verification with one-dimensional problems has been completed (Lee et al., 1993) and further verification efforts were completed in 1998.

The distinct models in the code employ a common set of utility routines and input file format. The various models are essentially isolated from each other; hence, future models can be added without affecting existing models. This also allows for ease in code maintenance and incorporation of future enhancements. Global variables in the code are virtually nonexistent. The code is written principally in the C language. Input data is in the form of that used by the lisp

language. An internal Lisp interpreter for the Scheme dialect of Lisp is part of the simulator whose purpose is to read the input data file and the internal data files containing default input data values. It also performs data checking.

Each module has its own user's manual documenting the data input specific to each module. Non-module-specific input are also given in the user's manuals, but with only the most commonly used input options covered. This reference manual describes the options for the non-module-specific data input in more detail and, also, describes the general numerical algorithms used.

For each simulation run, NUFT requires a text file to specify the input parameters (e.g., the grid definition and the hydrologic and fluid properties). This section describes the syntax of this file. The file format is free format (i.e., it does not matter what column the data is in, nor does it matter if data is continued past the current line or lines).

Input consists of lists of *data blocks*, or *data units*. Each data unit starts with a left parenthesis and ends with a right parenthesis. A data unit is of the following general form:

```
( <name> <data> <data> . . . )
```

where

<name> refers to the input “variable” that is being set or specified

<data> are items that are *real numbers*, *integer numbers*, *time real numbers*, *strings*, *pattern strings*, *words*, or other data units, or *list(s)* of data items.

The different data types are defined later in this section.

An alternate form for a data unit is (*<name> <data> <data> . . . \<name>*). An advantage of this form is that the model can more reliably tell the user the exact location of any unmatched parentheses.

Example:

```
(porosity 0.2)
(file-name "input.data")
(par 0.1 0.3 0.6)
```

This example sets three different variables. It sets the variable porosity to the numeric value, 0.2, the variable file-name to the string, "input.data", and the variable par to a list of three numeric values, 0.1 0.3 0.6.

Example:

```
(\rocktab (silt (porosity 0.3) (Kx 1.e-4) (Ky 1.e-4) (Kz 1.e-4))
          (sand (porosity 0.2) (Kx 1.e-2) (Ky 0.0) (Kz 0.0))
          (clay (porosity 0.4) (Kx 1.e-6) (Ky 0.0) (Kz 0.0)))
\rocktab)
```

This example shows how a data unit sets the variable, rocktab, to a list of data units using the alternate form.

Comment Character: Semi-colons in the input file serve as comment characters. That is, all characters on a given line after a semicolon are ignored by the program. Using comments is a good way for the user to annotate an input file. Using two semicolons instead of a single one is a good way to make sure that comments stand out.

Example:

```
(porosity 0.2) ;; this is how we set the value of porosity to 0.2
```

Units: All quantities are in MKS units (i.e., meters, kilogram, seconds). Thus, *hydraulic conductivities* are in meters/second, and *head* is in meters (see Table 1). Unitless quantities such as saturation, porosity, and concentrations are always fractional (i.e., between 0 and 1, inclusive), *not* percentages.

Table 1. Table of Units Used in Input to Models

length	<i>meters (m)</i>
mass	<i>kilograms (kg)</i>
time [†]	<i>seconds (s)</i>
temperature	<i>centigrade (°C)</i>
area	<i>m²</i>
volume	<i>m³</i>
mass density	<i>kg/m³</i>
molar density	<i>mole/m³</i>
permeability	<i>m²</i>
hydraulic conductivity	<i>m/s</i>
flow velocity	<i>m/s</i>
force	<i>Newton (Nt=kg·m/s²)</i>
pressure	<i>Pascals (Pa=Nt/m²)</i>
head	<i>m</i>
energy	<i>Joule (J=Nt·m)</i>
specific energy	<i>J/kg</i>
mass flux	<i>kg/s</i>
molar flux	<i>mole/s</i>
volumetric flux	<i>m³/s</i>
energy flux	<i>Watts (W = J/s)</i>
thermal conductivity	<i>W·m/°C</i>
dynamic viscosity	<i>Nt·s/m²=kg/m·s=10³ centipoise</i>
molecular diffusivity	<i>m²/s</i>

[†] model can accept other time units by the use of unit designators

Legal Data Types: Following are descriptions of valid data types:

- A *string* —any sequence of visible characters delimited by double quotes ““”, for example,
`"hello there"`
`"run3-B (test#2)"`

Note that spaces and parentheses are allowed in a string.

- An *integer number*—for example, 11

- A *real number* that is fixed or floating point—for example 1.23, -4.5e7, or 900.2E7. Note that **D or d exponents in the manner of FORTRAN are not allowed!**
- A *time number* that is a real number but that has the following unit designators as the last letter to denote units of time. This type of number is used to specify a time.

s	seconds
m	minutes
h	hours
d	days
M	months
y	years

If no unit designator is present, “seconds” is assumed.

Examples:

20.0	20 seconds
23.1s	23.1 seconds
45e4M	45e4 months

There must be no spaces between the number and the unit designator.

- A *word*—a sequence of nonblank, visible characters. A word can be a variable or may be used in the same way as a string except that it cannot have internal blanks. The model treats the words and strings as being distinct data types; the correct one, as specified in the documentation, is required.
- A *pattern string* —a special type of a string with the two unix shell-type “wild” characters
 - * and ?

so that a pattern string can represent an entire class of strings that matches the string pattern. The character * in a pattern matches any sequence of characters. Hence, the pattern “*” matches all strings. The character ? in a pattern matches any single character. Hence, the pattern “?” matches all strings with exactly one character.

Other Examples:

- The pattern “ex*” matches all strings that begin with the characters “ex”.
- The pattern “ex*b2*z” matches all strings that begin with “ex”, that are followed by any number (including zero) of strings that are then followed by the string “b2”, and that end with the string “z”.
- The pattern “r2?xay” matches all strings that begin with “r2”, followed by a single character, and that are then followed by the characters “xay”.

Include statement: The include statement is of the form

```
(include "<file-name>")
```

It is used to insert the contents of the file with the name "<file-name>" into the input file. The file must lie in the working directory under which NUFT is being run. It can only be used to replace a complete list (i.e., must be either a collection of data delimited by a closed set of parentheses or a single data item such as a number or string). For example, if the file "data1.inc" contains

```
(field (format list) (range "") (variables S1) (file-ext ".S1")
      (outtimes 0 70m 102m 222m 287m 342m 23h)
    )
```

and the file "data2.inc" contains the single entry

```
200m
```

then the following input data

```
(output
  (include "data1.inc")
  (forcetimes (outtimes (include "data2.inc") 201m))
)
```

will be interpreted by the model as equivalent to

```
(output
  (field (format list) (range "") (variables S1) (file-ext ".S1")
        (outtimes 0 70m 102m 222m 287m 342m 23h)
      )
  (forcetimes (outtimes 200m 201m))
)
```

The following is an example of an error. Suppose the file "file.inc" contains

```
(outtimes 0 70m 102m 222m
```

and the input file as the data item

```
(output
  (field (format list) (range "") (variables S1) (file-ext ".S1")
        (include "file.inc") 287m 342m 23h)
      )
  (forcetimes (outtimes 200m 201m))
  (file-ext ".S1")
)
```

This is an error because only complete lists or a single entry can be included (not to mention the fact that the parentheses will not match in the input file).

Include package statement: The `include-pkg` package statement is of the form

```
(include-pkg "<file-name>")
```

This statement is identical to the `include` statement except that it includes a file from the subdirectory that contains the NUFT executable rather than a file from the current working directory. The main purpose of the `include` package statement is to include a "package" of predefined input parameters that comes with the NUFT software distribution

Macro commands: Macro commands start with the character "#". There are three commands available: `#define`, `#ifdef`, and `#ifndef`. The following command defines a macro variable,

```
(#define <variable>)
```

Currently, a variable cannot be defined to be any particular value; it is used in conjunction with the other macro commands. The statements within

```
(#ifdef <variable> . . . .)
```

will be read as part of the input stream if `<variable>` is defined by the `#define` command. However, the statements within

```
(#ifndef <variable> . . . .)
```

will not be placed as input if `<variable>` is not defined. The `#define` statements must be in the same parenthesis level as, for example, `bctab`, `genmsh`, etc. The `#ifdef` and `#ifndef` commands can be placed anywhere, except that the body of statements in the conditional commands must be complete lists (i.e., parentheses match inside the macro command). Currently, the macro commands only work inside an input set for a module or inside `common`.

3. How to Read the Input Documentation

The documentation of the input data to the code is written with special symbols that are not actually part of the data input but that are used as convenient shorthand to mean certain things.

Following is a list of special symbols that are used:

- Any word starting with the symbol < and ending with >
- The symbols: | | . . . { } []

The meaning of these symbols is given as follows:

- Any italicized word starting with the character < and ending with the character > represents *data*, as described in the previous section, and will be called a *data token* (or *token*, for short).
- . . . is an abbreviation that means that more data items follow, but they are not specified at this point; further explanation of the required missing data items will follow.
- [] means that data items inside [] are optional; for example, [(xyz <real>)] means that the input value of variable xyz is optional.
- | | represents a logical “exclusive or” of two sets of data items; for example, (xyz <real>) | | (abc <real>) means that the user *must* specify either the variable xyz or abc but not both.
[(xyz <real>) | | (abc <real>)] means that the user has the *option* of either specifying xyz or abc.
- { } denotes a grouping of data items, usually used in conjunction with | | ; for example:
(xyz <real>) | | { (abc <real>) (ijk <integer>) } means that the user must either specify xyz or specify both abc and ijk.

In the input documentation, the following data tokens have special meanings:

<string>	a string
<integer>	an integer number
<real>	a real number
<t-real>	a time real number; the last character is alphabetic and denotes the units of time
<word>	a symbolic word
<pattern>	a pattern string

These data types are described in Section 2 of this reference manual.

4. Basic Elements of the Input File

Before going further, the user should have read the previous sections explaining the abbreviations and special symbols used in this input documentation.

General Form of Input Data

```
(<name-of-model>
  (title . . .)           ;; run title
  (outputfile . . .) ||   {(output-prefix . . .) (output-ext . . .)};; output file name
  (meshfile . . .) ||   (genmsh . . .);;mesh specification
  (time . . .)           ;;initial time
  (tstop . . .)           ;;ending time
  (dt . . .)              ;;initial time step
  (dtmax . . .)           ;;maximum time step
  (stepmax . . .)           ;;maximum no. of time steps
  (read-restart . . .)     ;;read from restart file
  (state . . .)           ;;set initial conditions
  (rocktab . . .)           ;;soil property type
  [ (grav-factor . . .) ]  ;;factor multiplying gravity vector

  [ (output . . .) ]       ;;output specification
  [ (srctab . . .) ]       ;;source tables
  [ (bctab. . .) ]         ;;boundary condition tables
)
```

Recall that the “. . .” denote subsequent data items that are to be explained later and that all of the input line past a semicolon is not read by the program but is for placing comments into the input file. The above data units do not have to occur in any particular order. The data entry `<name-of-model>` refers to the name of the model that is being used. For example, `uslp` refers to the single-phase unsaturated flow model.

Note that the use of the square brackets around `grav-factor`, `(output . . .)`, `(srctab . . .)`, and `(bctab . . .)` denote that these data units are optional. More optional data items will be described in subsequent text, but the ones shown above are the most likely to be used. Initial conditions are set either using `read-restart` or `state`. One, but not both, of these initial conditions must be present.

The preceding applies to NUFT modules that solve for flow and transport simultaneously. Some NUFT modules have the option of solving flow and transport sequentially. That is, the code first solves for the flow of phases, and then the transport equation for the contaminant(s) is solved at

each major time cycle in an alternating fashion. (If transport takes place at a much shorter time scale than does flow, the transport may take several time steps in a single, major time cycle.)

The form used when flow and transport are solved sequentially is as follows:

```
(common
  (title. . .)
  (outputfile . . .) || {(output-prefix. . .) (output-ext . . .)}
  (meshfile . . .) || (genmsh . . .)
  (time . . .)
  (tstop . . .)
)

(<name-of-flow-model>
  (dt . . .)
  (dtmax . . .)
  (stepmax . . .)
  (state . . .)
  (rocktab . . .)

  [ (output . . .) ]
  [ (srctab . . .) ]
  [ (bctab . . .) ]
)

(<name-of-transport-model>
  (dt . . .)
  (dtmax . . .)
  (stepmax . . .)
  (state . . .)
  (rocktab . . .)

  [ (output . . .) ]
  [ (srctab . . .) ]
  [ (bctab . . .) ]
)
```

Note that the flow model and transport model each has its own initial and maximum time steps and other data. Any input data that is common to both models are placed in the (common . . .) data unit. Any of the items in the common data unit can also appear in the data unit of the particular model, but they will be overridden by any specification that appears in the common data unit.

5. Input Data Documentation

The items in the input data file are classified in the following categories.

Mesh Generation Parameters

```
(genmsh . . .)  
(mesh-file . . .)
```

Time Stepping and Numerical Solution Parameters

```
(time . . .)  
(tstop . . .)  
(dt . . .)  
(dtmax . . .)  
(stepmax . . .)
```

Output Specification

```
(title . . .)  
(output . . .)
```

Specification of Initial Conditions

```
(state . . .)  
(read-restart . . .)
```

Rock Property Specification

```
(rocktab . . .)
```

Source Term Specification

```
(srctab . . .)
```

Boundary Condition Specification

```
(bctab . . .)
```

Other options

```
(upstream-weighting . . .)  
(include . . .)
```

5.1 Mesh-Generation Parameters

NAME

genmsh

Internally generate a rectangular or cylindrical grid system

SYNOPSIS

```
(genmsh
  (coord <coord-type>)
  (down <num> <num> <num>)
  (dx <numx-0> <numx-1> . . .)
  (dy <numy-0> <numy-1> . . .)
  (dz <numz-0> <numz-1> . . .)
  (mat
    (<el-name-prefix> <mat-type>
      <i0> <i1> <j0> <j1> <k0> <k1>)
      . . .
    (<el-name-prefix> <mat-type>
      <i0> <i1> <j0> <j1> <k0> <k1>)
  )
  [ (isot-dir) ]
  [ (isot
      (<num> <dir> <I0> <II> <j0> <j1> <k0> <k1>)
      . . .
      (<num> <dir> <I0> <II> <j0> <j1> <k0> <k1>)
    ) ]
  [ (volfac
      (<num> <I0> <II> <j0> <j1> <k0> <k1>)
      . . .
      (<num> <i0> <i1> <j0> <j1> <k0> <k1>)
    ) ]
  [ (areafac
      (<num> <dir> <I0> <II> <j0> <j1> <k0> <k1>)
      . . .
      (<num> <dir> <I0> <II> <j0> <j1> <k0> <k1>)
    ) ]
  [ (write-mesh "<file-name>" ) ]
  [ (write-grid "<file-name>" ) ]
  [ (write-gdef "<file-name>" ) ]
```

```
[ (gdef-ext "<file-ext>" ) ]  
[ (non-log) ]  
[ (wrap-around) ]  
)
```

NAME

mesh-file

Allow user to easily generate a regular rectangular or cylindrical mesh (for more specific meshes, see the `mesh-file` option)

PARAMETERS

`(coord <coord-type>)`

set type of coordinate system

`<coord-type>`

coordinate system type, options are

`rect`

`cylind`

`(down <numx> <numy> <numz>)`

sets the components of the vector pointing downward in the direction of the gravity vector. The program will internally normalize the vector to unity. Setting the components all to zero will turn off gravity in the model. The vector is always with respect to a rectangular coordinate system (X, Y, Z). For a rectangular mesh, the coordinate system coincides with the rectangular coordinate system (x, y, z) of the mesh. If the mesh is cylindrical, the vector is with respect to a coordinate system (X, Y, Z) where X is the axis defined by $\theta = 0, z = 0$; Y is the axis defined by $\theta = 90^\circ, z = 0$, and the axis Z is defined by $r = 0$.

`(dx <numx-0> <numx-1> . . .)`

`(dy <numy-0> <numy-1> . . .)`

`(dz <numz-0> <numz-1> . . .)`

sets the mesh subdivisions in the x, y, and z coordinate directions.

Numbers that are repeated can be abbreviated; for example, $3*5.0$ would stand for three repeats of the numeral 5 (i.e., 5.0 5.0 5.0).

```
(mat
  (<el-name-prefix> <mat-type>
    <i0> <i1> <j0> <j1> <k0> <k1>)
    . . .
  (<el-name-prefix> <mat-type>
    <i0> <i1> <j0> <j1> <k0> <k1>)
)
)
```

sets material property name of each element.

The element names will be of the form $<\text{el-name-prefix}>\#<i>:<j>:<k>$ where $<i>$, $<j>$, $<k>$ denote the i , j , and k indices. The material type $<\text{mat-type}>$ is defined in `(rocktab . . .)`. The symbols `nx`, `ny`, and `nz` can be used anywhere in place of numbers where an index is required. The model interprets these to mean the number of subdivisions in the x, y, and z directions, respectively.

[`(isot-dir)`]

This parameter is optional. It affects the choice of the permeability (or hydraulic conductivity) parameter. See the documentation of the permeability parameters `K0`, `K1`, and `K2` in subsequent text. This parameter should not be present for models where isotropic permeability is desired.

[`(isot`

```
(<num> <dir> <I0> <II> <j0> <j1> <k0> <k1>)
  . . .
(<num> <dir> <i0> <II> <j0> <j1> <k0> <k1>)
```

)]

sets `isot` = 0,1,2 in x,y,z directions, respectively; default is `isot` = 0 for all elements. The parameter `isot` selects which of the permeability (or hydraulic conductivity) values `K0`, `K1`, `K2` set in `rocktab` are used for the particular element.

[`(volfac`

```
(<num> <i0> <i1> <j0> <j1> <k0> <k1>)
  . . .
(<num> <i0> <i1> <j0> <j1> <k0> <k1>)
```

)]

sets volume modifying factor. Multiplies the volume of the specified elements by `<num>`

```

[ (areafac
  (<num> <dir> <i0> <i1> <j0> <j1> <k0> <k1>)
  . . .
  (<num> <dir> <i0> <i1> <j0> <j1> <k0> <k1>)
) ]
  sets area modifying factor
  <dir>
    valid options: x, y, or z;
    dir = x will mult. the area between block i,j,k and i+1,j,k
    dir = y will mult. the area between block i,j,k and i,j+1,k
    dir = z will mult. the area between block i,j,k and i,j,k+1

[ (write-mesh "<file-name>") ]
  write out mesh information using $con and $elc format

[ (write-grid "<file-name>") ]
  write out mesh information using $freegrid format

[ (write-gdef "<file-name>") ]
  write out minimal geometry information about the grid to file; the format of the grid
  definition file is
    $gdef
    $type    <word>    mesh type specified in coord
    $nx      <real>     no. of subdivisions in x direction
    $ny      <real>     no. of subdivisions in y direction
    $nz      <real>     no. of subdivisions in z direction
    $order   <word>     ordering of elements (e.g., xyz, yzx, or zxy
    $dx      <real>     subdivisions in first coordinate
    <real>
    . . .
    <real>
    $dy      <real>     subdivisions in second coordinate
    <real>
    . . .
    <real>
    $dz      <real>     subdivisions in third coordinate
    <real>
    . . .
    <real>

```

Line breaks are treated as significant in this format.

[(gdef-ext "<file-ext>")]

Write out minimal geometry information about the grid to file with file extension.

[(non-log)]

By default, flow areas in the radial direction for cylindrical coordinates are calculated using the logarithmic formula.

$$A = \frac{1}{2}(\Delta r_{i-1} + \Delta r_i)\Delta\theta\Delta z / \ln(r_i / r_{i-1})$$

If the non-log flag is present, flow areas are calculated as

$$A = (r_i - \Delta r_i / 2)\Delta\theta\Delta z$$

[(wrap-around)]

If *<coord-type>* is set to cylind, and the angles in the y (angular) direction sum to 360°, the elements at $j = 1$ are adjacent to corresponding elements at $j = ny$ where ny is the number of subdivisions in the y direction. When this option is present, the model will make connections between these elements. This option is valid only for *<coord-type>* set to cylind. Default is no wrap-around. (Note that the wrap-around option will increase the bandwidth of the matrix when using the direct-solution option or the comb option of the preconditioned conjugate gradient method and, therefore, will increase cpu time for these methods.)

NOTES

- Indices start from 1.
- Coordinates of grid blocks sides start from zero and are incremented by corresponding values in (dx . . .), (dy . . .), and (dz . . .).
- The first grid block center in x direction has $x = <dx0>/2$, where $<dx0>$ is the first number in (dx . . .).
- If coordinate type is cylind, the first coord x is radial distance, the second y is angle, and third z is longitudinal to central axis.
- The units of the angles in the cylind option are in degrees.

NAME

mesh-file

Specify name of mesh file

SYNOPSIS

```
(mesh-file "<file>" )
```

DESCRIPTION

The user can set up a grid either by using the (genmsh . . .) option or by generating a mesh file outside the program and then reading the mesh file into the NUFT model. The genmsh option can only produce grids that are rectangular or cylindrical. The advantage of a mesh file is that the user can write a program to generate the user's own grid, taking full advantage of the generality of the integrated, finite-difference method. The format of the mesh file is described in Appendix A.

PARAMETERS

<file> name of mesh file

NAME

grav-factor

Set gravity modification factor (optional), including gravity orientation (optional)

SYNOPSIS

(**grav-factor** <*factor*>)

DESCRIPTION

The user can optionally multiply the gravity vector in the model by this factor. (If the mesh-file option is used, this factor is multiplied in addition to the beta factor read from the mesh file.) By setting the factor to zero, gravity may be turned off. By setting this factor to the cosine of the angle of inclination relative to the vertical downward direction assumed when creating the mesh file, one can change the orientation of the model without rereading the mesh file. When using the genmsh option, one can do the same thing through the down vector.

The default value is unity.

PARAMETERS

<*factor*> real number multiplying gravitational acceleration
vector (default: 1.0)

5.2 Time-Stepping Parameters

NAME

time, tstop, dt, dtmax, stepmax

Set automatic time-stepping parameters

SYNOPSIS

```
(time <t-real>)
(tstop <t-real>)
(dt <t-real>)
(dtmax <t-real>)
(stepmax <integer>)
```

DESCRIPTION

These input parameters are related to the automatic determination of the time-step size. Except for `stepmax`, their values are *<t-real>* (i.e., *time-real*). The time-stepping algorithm that is used is described in Appendix B. These parameters are required. Section 5.2.2 describes optional parameters related to the automatic time-stepping and Newton Raphson iteration.

PARAMETERS

time	initial simulation time
tstop	simulation stopping time
dt	initial time step
dtmax	maximum time step allowed
stepmax	maximum number of time steps that, if exceeded, will halt the run

NOTES

The initial simulation time step overrides the time read in from a restart file if the `restart` command is present. If the `time` command is not present, the time read in from the restart file will be used.

NAME

tolerdt, reltolerdt, dtmin

Set automatic time-stepping parameters (optional)

SYNOPSIS

```
(tolerdt <real> <real> . . .)  
(reltolerdt <real> <real> . . .)  
(dtmin <t-real>)
```

DESCRIPTION

These input parameters are related to the automatic determination of the time-step size at each time step, as generated by the time-stepping algorithm. The algorithm seeks to control maximum changes between the components of the solution vector from the current time step to the next. The algorithm used is described in Appendix B.4.

PARAMETERS

tolerdt maximum tolerance for change in components of solution vector from one time step to the next

The data parameters are in the form (*<variable> <real>*) values, one for each type of quantity in the solution vector. For example, for a model with saturation and pressure in Pascals as the primary variables would have the form

```
(tolerdt (S 0.5) (P 1.e5))
```

See the specific model documentation for specific details and the default values.

reltolerdt maximum relative tolerance for change in components of solution vector from one time step to the next

The data parameters are of the form of (*<variable> <real>*) values, one for each type of quantity in the solution vector. For example, for a model with saturation and pressure as the primary variables, in that order, the following

```
(reltolerdt (S 0.5) (P 0.2))
```

would seek to adjust the time step such that the saturation does not change more than 10% nor the pressure more than 20% relative to the previous time-step values. The model will use the larger of the two time steps calculated from the **tolerdt** and the **reltolerdt** values. See the specific model documentation for specific details and default values.

dtmin minimum time step (default 0.0)

5.3 Parameters for Numerical Methods

NAME

tolerconv, reltolerconv, itermax, iterbreak, cutbackmax

Set parameters controlling Newton-Raphson iteration

SYNOPSIS

```
(tolerconv <real> <real> . . .)  
(reltolerconv <real> <real> . . .)  
(itermax <integer>)  
(iterbreak <integer>)  
(cutbackmax <integer>)
```

DESCRIPTION

Parameters controlling Newton-Raphson iteration (see Appendix B for description).

PARAMETERS

tolerconv Maximum tolerance for change in components of solution vector from one Newton-Raphson (NR) iteration to the next during a time step for NR convergence criteria to be satisfied

The data parameters are of the form of (*<variable> <real>*) values, one for each type of quantity in the solution vector. For example, a model with saturation and pressure in Pascals as the primary variables, in that order, would have the form

```
(tolerconv (S 0.5) (P 1.e5))
```

See the specific model documentation for specific details and the default values. Numbers that are repeated can be abbreviated; for example, $3*0.1$ would stand for three repeats of the numeral 0.1 (i.e., 0.1 0.1 0.1).

reltolerconv Maximum relative tolerance for change in components of solution vector from one NR iteration to the next for NR convergence

The data parameters are of the form of a list of nonnegative *<real>* values, one for each type of quantity in the solution vector. For example, for a model with saturation and pressure as the primary variables, in that order, the following

```
(reltolerconv (S 0.5) (P 0.2))
```

would specify that NR convergence criteria is met if the saturation does not change more than 10% and if the pressure changes no more than 20% relative to the previous NR iteration values. The convergence criteria of the model are satisfied if one or both of the **tolerconv** and

	<p><code>reltolerconv</code> criteria are satisfied. See the specific model documentation for details and default values.</p>
<code>itermax</code>	<p>Maximum NR iterations</p> <p>If exceeded (i.e., if NR convergence has not been reached, and the NR iteration is greater than this number), the time-step size is cut back by one-half, and the time step is started over (default value: 8).</p>
<code>iterbreak</code>	<p>Go on to next time step if this many NR iterations have been reached, regardless of whether the NR convergence criteria are met (default value: 1000000000).</p>
<code>cutbackmax</code>	<p>Maximum number of times in a given time step that the time step has to be started over again due to lack of NR convergence</p> <p>If exceeded, the program will print an error message and then stop (default value: 100).</p>

NAME

linear-solver, pcg-parameters

Set parameters for linear-solver

SYNOPSIS

```
(linear-solver <word>)

[ (pcg-parameters (precond <word>) (north <integer>)
                  (toler <real>) (itermax <integer>)
                  [ (direct <integer> <integer> . . . ) ]
                )
  ]
[ (ilu-degree <integer>) ]
```

DESCRIPTION

These parameters are for the solution of linear system of equations that needs to be solved in each iteration of the Newton-Raphson method.

Example:

```
(pcg-parameters (precond comb) (north 10) (toler 1.e-4)
                (itermax 30) (direct 1 0))
```

PARAMETERS

linear-solver Sets the method used to solve the system of linear equations in the Newton-Raphson method

Valid values are

lublknd	standard ordered, constant bandwidth; block-banded gaussian elimination (default)
vband	variable bandwidth elimination with reverse cuthill-mckee bandwidth minimization
d4vband	d4 ordered, variable bandwidth elimination
pcg	orthomin preconditioned conjugate gradient method

(default value: blknd)

pcg-parameters Parameters for preconditioned conjugate gradient (PCG) method

precond Type of preconditioning method used

Options:

dkr	first degree incomplete ILU
ilu	incomplete ILU decomposition with variable fill-in
d4	incomplete ILU with d4 ordering with variable fill-in
comb	combinative method

bgs block gauss-seidel method

none no preconditioning

The recommended option for problems with linear or nearly linear equations (e.g., ucsat) is

```
(pcg-parameters (precond bgs)
(toler 1.e-3) (itermax 500) (north 20))
```

For problems with intermediate nonlinearity (e.g., us1p, usnt), the incomplete ilu of degree 0 (no fill-in) with generalized D4 ordering is recommended,

```
(pcg-parameters (precond d4) (toler 1.e-3)
(itermax 200) (north 15))
```

For problems with severe nonlinearities (e.g., usnt with phase changes or highly nonlinear characteristic curves)

```
(pcg-parameters (precond d4) (toler 1.e-3)
(itermax 200) (north 15)) (ilu-degree 1)
```

The degree of incomplete LU decomposition is set by ilu-degree to 1. Recall that the default is 0.

Note that the D4 ordering is generalized, meaning that it also works when connections between elements are nonstandard with respect to the standard i,j,k lattice.

north number of orthogonalizations performed in the orthomin PCG method

toler convergence tolerance for the PCG method

The convergence criteria is that the norm of the residual vector be less than toler times the norm of the initial residual vector. The mean square norm is used

$$\| \mathbf{r} \| = \left(\frac{1}{n} \sum_{i=1}^n r_i^2 \right)^{1/2}$$

itermax maximum number of PCG iterations

If exceeded, the time-step size will be cut in half, and the time step will be started over.

direct specifies which of the balance equations are solved directly in the combinative method used only for the (preconditioning option comb)

This is needed only for comb preconditioning method. Parameters are a list of <integer>s, one for each balance equation. A nonzero value will cause the linear equations for that balance equation to be solved directly by gaussian elimination.

ilu-degree degree of fill-in for ILU decomposition

Degree 0 equals no fill-in.

NAME

time-method

Specify the time discretization method

SYNOPSIS

```
(time-method <string>)
```

DESCRIPTION

This parameter specifies whether to use either a fully implicit or fully explicit discretization in time.

PARAMETERS

time-method

Valid values are "fully-implicit" or "explicit" (default value: model-specific).

NAME

upstream-weighting

Set upstream-weighting

SYNOPSIS

```
[ (upstream-weighting <weight>) ]
```

DESCRIPTION

sets upstream weighting

To calculate the advective flux $c\mathbf{V}$ of a component between two connected elements, the model will use the weighting

$$c = a c_{up} + (1 - a) c_{down}$$

based on the weighting factor w where $a = wL_{up}/(wL_{up} + (1-w)L_{down})$ where L refers to flow lengths, the subscript *up* refers to upstream value, and the subscript *down* to downstream value. The default is $w=1$, which is full upstream-weighting.

PARAMETERS

<weight> weighting factor: usually between 0 and 1 inclusive (default: 1.0)

NAME

flux-correction, flux-correct-options

Set parameters controlling flux-correction scheme

SYNOPSIS

```
(flux-correction <word>)
  (flux-correct-options (method <word>) (iter <integer>))
```

DESCRIPTION

By default, the model uses complete upstream-weighting of mobilities to numerically calculate the advective flux between two elements (see upstream-weighting for changing the amount of upstream-weighting). An alternative method is to use the flux-correction scheme of Smolarkiewicz [1983], which is a modification of the upstream-weighting. If `flux-correction` is set to `on`, the method of Smolarkiewicz is used with three iterations. The number of iterations can be changed by setting parameters in `flux-correct-options`. An upstream, modified harmonic-averaging method can also be used instead of the Smolarkiewicz.

PARAMETERS

`flux-correction` can be set to either `on` or `off` to turn on flux-correction on or off
`method` choice of flux correction method

`smolark` flux-correction using Smolarkiewicz's method

`harmonic` flux-correction using harmonic averaging

`iter` number of iterations for flux-correction

Using more than two iterations for the harmonic method can lead to spatial oscillations in the solution. Using value of 1 for the harmonic appears to give the same results as does the Smolarkiewicz method.

NAME**title**

Set run title

SYNOPSIS

```
(title <string>)
```

DESCRIPTION

This input parameter specifies the run title, which is placed at the top of output files.

Example:

```
(title "Run 3A: hydrological study")
```

5.4 Output Specifications

NAME

output-file, output-prefix, output-ext

Specify names of the various output files

SYNOPSIS

```
[ (output-file <string>) ]  
[ (output-prefix <string>) ]  
[ (output-ext <string>) ]
```

DESCRIPTION

The model will place its output into various files. At least one main output file, and possibly, several auxillary files, will be generated by the `output` option described in this section. The user has two possibilities for specifying the names of these files: (1) separately name each file using the `output-file` parameter to name the main output file and the `file` parameter in `output` to name the auxilliary files; (2) more conveniently, have all files use the same prefix (e.g., the run name), and use different suffixes for each file (e.g., “.out,” “.pg,” “.T”). If the latter method is used, the prefix is set using `output-prefix`, and the suffix of the main output file is set using `output-ext`. The suffix for the names of the auxilliary files is set using `file-ext`, described in this section, on the data unit `output`.

PARAMETERS

<code>output-file</code>	name of main output file
<code>output-prefix</code>	All output files, including those written by the <code>(output . . .)</code> data field, will have this prefix; this is usually used to specify a single run name where all output files start with this name (default: prefix of input file name).
<code>output-ext</code>	Instead of specifying the output file, one can specify the suffix of the output file (default: “.out”).

NAME

output

Specify output

SYNOPSIS

```
(output
  (field
    [ (file "<file-name>" )
      (format <options>)
      {(range "<element range>" . . .) | |
       (index-range (<i0> <i1> <j0> <j1> <k0> <k1>) . . .)}
      (variables <el-var0> <el-var1> . . .)
      (outtimes <t0> <t1> . . .)
    )
  )
  (flux-field
    [ (file "<file-name>" )
      (format <options>)
      {(crange ("<element range>" "<element range>" ) . . .) | |
       (index-crange ((<i0> <j0> <k0> <i1> <j1> <k1>) . . .))}
      (variables <con-var0> <con-i> . . .)
      (outtimes <t0> <t1> . . .)
    )
  )
  (history
    (variable <el-var>)
    (element "<element name>" )
    [ (file "<file-name>" )
    )
  )
  (flux-history
    (variable <con-var0>)
    (comp-flux (<comp> <phase>)) )
    {(connection "<element name>" "<element name>" ) | |
     (index-con <i0> <j0> <k0> <i> <j> <k>) }
    [ (file "<file-name>" )
    )
  )
  (srcflux
    (name <src-name>)
    (comp <comp-name>)
    [ (file "<file-name>" )
      (outtimes <t0> <t1> <t2> . . .)
    )
  )
)
```

```

        [ (cumulative) ]
    )
(bcflux
    (name <bc-name>)
    (comp <comp-name>)
    [ (file "<file-name>" ) ]
        (outtimes <t0> <t1> <t2> . . .)
    [ (cumulative) ]
)
(forcetimes
    (outtimes <t0> <t1> <t2> . . .)
)
(restart
    [ (file "<file-name>" ) ]
        (outtimes <t0> <t1> <t2> . . .)
)
(extool
    (range "<element range>" . . .) ||
    (index-range (<i0> <i1> <j0> <j1> <k0> <k1>) . . .)
    (variables <var0> <var1> . . .)
    (outtimes <t0> <t1> <t2> . . .)
    [ (file "<file-name>" ) ]
)
)

```

DESCRIPTION

Specifies the writing of various output data to files

Any of the preceding options can be specified; none of them is required.

PARAMETERS

```

(field
    [ (file "<file-name>" ) ]
    (format <options>)
    {(range "<element range>" . . .) ||
    (index-range (<i0> <i1> <j0> <j1> <k0> <k1>) . . .)}
    (variables <var0> <var1> . . .)
    (outtimes <t0> <t1> <t2> . . .)
)

```

Outputs element data

```

(flux-field
  [ (file "<file-name>" ) ]
  (format <options>)
  { (crange ("<element range>" "<element range>" . . .) || 
    (index-crange ((<i0> <j0> <k0> <i1> <j1> <k1>) . . .)) }
  { (phase-fluxes <phase0><phase1> . . .) || 
    (comp-fluxes (<comp0> <phase0>) (<comp1> <phase1>) . . .) }
  (outtimes <t0> <t1> <t2> . . .))
)

Outputs flux output data

(history
  (variable <var. name>)
  (element "<element name>" )
  [ (file "<file-name>" ) ]
)

Specifies output of element variable vs. time

(flux-history
  (phase-flux <phase>) || 
  (comp-flux (<comp> <phase>)) 
  { (connection "<element name>" "<element name>" ) || 
    (index-con <i0> <j0> <k0> <i1> <j1> <k1>) }
  [ (file "<file-name>" ) ]
)

Specifies output of flux variable vs. time

(srcflux
  (name <src-name>)
  (comp <comp-name>)
  [ (file "<file-name>" ) ]
  (outtimes <t0> <t1> <t2> . . .)
  [ (cumulative) ]
)

Outputs the total instantaneous flux of a component, <comp-name>, due to a source term
set in srctab

The name of the source term <src-name> is the desired one in srctab. The sign
convention is such that flow out of the problem domain is positive, and flow out of the
domain is negative. If the statement [ (cumulative) ] is present, the cumulative flux
is outputted instead of the instantaneous flux. Note that cumulative fluxes are reset to 0 at
the beginning of a restart.

```

```
(bcflux
  (name <bc-name>)
  (comp <comp-name>)
  [ (file "<file-name>" ) ]
  (outtimes <t0> <t1> <t2> . . .)
  [ (cumulative) ]
)
```

Outputs the total instantaneous flux of a component, *<comp-name>*, flowing into a set of elements specified as boundary elements in *bctab*

The name of the boundary condition *<bc-name>* is the desired one in *bctab*. The sign convention is such that flow out of the set of elements is positive, and flow into the elements is negative. If the statement [(cumulative)] is present, the cumulative flux is outputted instead of the instantaneous flux.

```
(forcetimes
  (outtimes <t0> <t1> <t2> . . .)
)
```

Forces the model to hit specified times without necessarily outputting any information
This is used to prevent the model from skipping over the times at which fluxes or boundary conditions change suddenly.

```
(restart
  [ (file "<file-name>" ) ]
  (outtimes <t0> <t1> <t2> . . .)
)
```

Write out a restart record at the specified times

Restart record can be read in as initial conditions using the *restart* command. Backup restarts are also written (see subsection on reading *restart* information). The *state* command must not be present when using this command.

```
(extool
  (range "<element range>" . . .) ||
  (index-range (<i0> <i1> <j0> <j1> <k0> <k1>) . . .)
  (variables <var0> <var1> . . .)
  (outtimes <t0> <t1> <t2> . . .)
  [ (file "<file-name>" ) ]
)
```

Outputs a "time-history file" in *extool* format program

NOTES

1. (outtimes <t0> <t1> <t2> . . .) can be replaced by either of the following:

a) (`outtimes`), which means all times

b) (`triggers`

```
( (wake <state-var> (range "<el-name>" . . .) <op> <val> )
  (cond <state-var> (range "<el-name>" . . .) <op> <val> ) )
  . . .
  ( (wake <state-var> (range "<el-name>" . . .) <op> <val> )
  (cond <state-var> (range "<el-name>" . . .) <op> <val> ) )
)
```

which checks to see, at every time step, if any of the triggers goes off. A trigger goes off if the condition `v <op> <val>` is true for the wake field where `v` is the value of the state variable `<state-var>` at the elements with names in the range (`"<el-name>" . . .`). If this condition is true, the corresponding condition for the `cond` field is checked; if true, then trigger goes off, and output occurs. Triggers only go off once.

(format `<options>`) : `<options>` can be any of following list or by-list list values and element names (contsac format):

<code>list</code>	list values and element names
<code>by-set</code>	list values as a lisp vector, [#n v1 v2 . . . vn]
<code>by-x</code>	list x coord. and value
<code>by-y</code>	list y coord. and value
<code>by-z</code>	list z coord. and value
<code>by-xyz</code>	list x, y, z coordinates and values
<code>by-ijk</code>	list i,j,k index and value
<code>by-xtable</code>	format compatible with <code>state</code> using by-xtable method; user needs to comment out output header
<code>by-ytable</code>	format compatible with <code>state</code> using by-ytable method; user needs to comment out output header
<code>by-ztable</code>	format compatible with <code>state</code> using by-ztable method; user needs to comment out output header
<code>tabular</code>	multicolumn format
<code>contour</code>	format readable by the <code>nvview</code> program for MS-DOS

1. In all options except `extool` option, if (`file . . .`) or (`file-ext . . .`) is missing, it will write to output file.
2. All `extool` options must write to a single file. Only the file name of the first `extool` data block is read; file name of the rest are ignored if present.
3. (`file "<file-name>"`)> can be replaced by (`file-ext "<file-name suffix>"`)> if the field output-prefix has been set; the output file name is then the suffix appended to the output-prefix.
4. i, j, k indices start from 1 and go to nx, ny, or nz, respectively.

OPTIONAL OUTPUT INFORMATION

(print-input on)	output input data
(print-pcg off)	output pcg convergence information
(print-mat-bal off)	output material balance
(print-matrix off)	output matrix
(print-accum off)	output accumulation
(print-accum-coefs off)	output accumulation coefficients
(print-flux off)	output internal fluxes
(print-flux-coefs off)	output internal flux coefficients
(print-sol off)	output solution at end of each time step
(print-sol-NR off)	output solution between NR iterations
(print-dsol off)	output changes between NR iterations
(print-NR-iter off)	output NR iteration nnumber to stderr
(print-qsrc off)	output source terms
(print-bc off)	output bc terms
(print-log off)	turn on printing to a log file
(print-all-terms off)	output values of terms
(print-eqtpar off)	output eqtpar structure
(print-timepar off)	output time step data
(print-srctab off)	output src tables
(print-bctab off)	output bc tables
(print-rocktab off)	output rock tables
(print-fprop off)	output fluid properties
(print-ifdgrid off)	output grid data
(print-dt-state off)	output time-step restriction
(print-conv-state off)	output NR convergence restriction

5.5 Specifying Initial Conditions

NAME

state

Set initial conditions for primary variables

SYNOPSIS

```
(state
  (<state var. name> <method> <data>)

  . . .
  (<state var. name> <method> <data>)

)
```

NOTES

This command must be left out when reading initial data from a `restart` file using the `restart` command.

Examples:

```
(<state var. name>) (by-key ( "<elem. range>" <value> )
  . . .
  ( "<elem. range>" <value> ))
(<state var. name> by-set <vector>)
```

where `<vector>` = [`#<n>` `<v0>` `<v1>` . . .]
and `<n>` is number of components (the `#<n>` field is optional).

Here, [and] are actual characters and do not represent delimiters of optional parameters.

```
(<state var. name> by-xtable <table>),
(<state var. name> by-ytable <table>), or
(<state var. name> by-ztable <table>)
```

where `<table>` = (`<z0>` `<val0>` `<z1>` `<val1>` . . .) is a table of values with respect to the appropriate x, y, or z coordinate

NOTES:

1. A state variable can appear more than once to overwrite values set by previous specifications. For example, values for all elements can be set by “by-xtable” or “by-set” and then “touched up” using “by-key”).”
2. The program will terminate if the primary variables have not been set for each of the elements.

NAME

read-restart

Read restart record

SYNOPSIS

```
(read-restart
  (file "<file-name>")
  [ (time <restart-time>) ]
)
```

DESCRIPTION

Sets initial conditions for primary variables by reading in a restart record from a file instead of using (state . . .)

PARAMETERS

"<file-name>" string

name of the restart file created by the option
(output . . . (restart . . .) . . .)

<restart-time>" real

Time used to search in the restart file; the first restart record with time = <restart-time> will be read in. The initial time of the simulation run is set by time and overwrites the time read in through the restart file. If time is not present, the initial of the run is set to the time read in.

NOTES

Restart files are created from a previous run using the restart option in the output command.

Restart Backup

The model will periodically write out restart records so that the model can be restarted in case of system failure. The model alternately writes out to two restart files named <prefix>.re0 and <prefix>.re1, where <prefix> is set by output-prefix. Each file contains a single record; previous records are overwritten. Two files, instead of a single one, are used to prevent losing a record if the system fails during a write. The user must check the two files to see which is the most recent. The model writes to a file at periodic intervals, based on the wall-clock time.

(backup <option>) optional

<option> word

If set to on, the model will periodically write backup restarts. If set to off, the model will not do backups. Default is on.

(backup-period <backup-time>) optional

<backup-time> t-real Wall-clock time period for model to perform backup restarts
Default value is 10m (i.e., 10 minutes).

5.6 Setting Rock Properties

NAME

rocktab

Set rock properties

SYNOPSIS

```
(rocktab
  (<rock-type name>
   . . .
  )
  . . .
  (<rock-type name>
   . . .
  )
)
```

DESCRIPTION

Sets rock properties for each rock type

PARAMETERS

<rock-type name> the name of the rock type

5.7 Setting Source Terms

NAME

srctab

Set source terms

SYNOPSIS

```
(srctab
  (compflux <comp-name>
    (range "<elem-range>" "<elem-range>" . . .)
    (table <flux-table>)
    [ (enthalpy <enthalpy-table>) ]
  )
  . . .
  (compflux
    . . .
  )
  (phaseflux <phase>
    (name <phaseflux-name>)
    (range "<elem-range>" "<elem-range>" . . .)
    (table <phase-flux-table>)
    (setcomp
      (<comp-name>
        (table <conc-table>)
        following only for thermal models
        [ (enthalpy <enthalpy-table>) ]
      )
      . . .
      (<comp-name> internal)
      . . .
      (<comp-name>
        . . .
      )
    )
  )
  . . .
  (phaseflux
    . . .
  )
)
```

DESCRIPTION

Specifies the component flux into an element or range of elements through a table of source fluxes at specified points in time.

NOTES

Linear interpolation is used for time intervals between the table values. Positive flux is flux *into* an element; negative flux is *out* of an element.

PARAMETERS

<comp-name> *word*

Name of the component (model-specific), which will be forced out or into the element(s)

<elem-range> *pattern string*

Range of elements that will have the source term flux specified by the flux table

<flux-table> *table*

Component mass flux table of the form

<t0> <q0> <t1> <q1> . . .

where the time values are given by *<t0>*, *<t1>*, . . ., which are of data type *t-real*, and the mass flux values are *<q0>*, *<q1>*, . . ., which are of data type *real* and are in units of kg/sec (except that us1p model uses volumetric flux m³/s). (See Section 6 for an important note.)

<phase> *name*

Name of the phase

<phaseflux-name> *name*

Name of the phase flux set

This is used by output options.

<phase-flux-table> *list of reals*

Table of mass fluxes of the phase that are of the form

<t0> <q0> <t1> <q1> . . .

where the time values are given by *<t0>*, *<t1>*, . . ., which are of data type *t-real*, and the mass flux at these times are *<q0>*, *<q1>*, . . ., which are of data type *real* and are in units of kg/sec. Linear interpolation is used for values between the time values. The last time must be greater than the end time of the run.

<conc-table> *list of reals*

Table of mole or mass fraction concentrations of the components within the phase stream; the table is of the form

<t0> <x0> <t1> <x1> . . .

where the time values are given by *<t0>*, *<t1>*, . . ., which are of data type *t-real*, and the concentrations are *<x0>*, *<x1>*, . . ., which are of data type *real*. An alternative is to specify that the component concentrations used for the flux are those of element itself; this is done by placing the command (*<comp> internal*) instead of (*<comp> (table <conc-table>)* . . .). Concentrations are in mass fraction if

(input-mass-fraction on) is present. Otherwise, concentrations are in mole fraction.

<enthalpy-table> list of reals

Table of specific enthalpies ("J"/kg) of the component

The table is of the form

$<t0> <h0> <t1> <h1> \dots$

where the time values are given by $<t0>$, $<t1>$, \dots , which are of data type *t-real*, and the enthalpies at these times are $<h0>$, $<h1>$, \dots , which are of data type *real* and have units of Joules/kg. Linear interpolation is used for values between the time values. The last time must be greater than the end time of the run.

internal

Calculate component and energy fluxes based on concentrations and enthalpies in the respective phase within the element instead of specifying through a table of concentrations and enthalpies

5.8 Specifying Boundary Conditions

NAME

bctab

Set boundary conditions

SYNOPSIS

```
(bctab
  (<bc-name>
    (range "<elem-range>" "<elem-range>" . . .)
    (tables
      (<primary-var> <var-table>)
      (<primary-var> <var-table>)

      . . .

    )
    [ (factor
      (<comp-name> <comp-factor-table>)
      (<comp-name> <comp-factor-table>)

      . . .

    ) ]
    [ (phasefactor
      (<phase-name> <phase-factor-table>)
      (<phase-name> <phase-factor-table>)

      . . .

    ) ]
  )
  . . .
  (<bc-name>
    . . .

  )
  . . .
  (<bc-name>
    (range "<elem-range>" "<elem-range>" . . .)
    (clamped)
    [ (factor
      (<comp-name> <factor-table>)
      (<comp-name> <factor-table>)

      . . .

    ) ]
    [ (phasefactor
```

```

        ( <phase-name> <phase-factor-table> )
        ( <phase-name> <phase-factor-table> )

        . . .

    ) ]
)
)

```

PARAMETERS

<bc-name> *word*

Name of the boundary condition

Each boundary condition has a distinct name used for identification by output options.

"*<elem-range>*" *pattern string*

Range of elements that will have the boundary condition

<primary-var> *word*

Name of the primary variable (model-specific) that is being specified by the associated table

<var-table> *table*

Table of primary variable values of form

<t0> *<var0>* *<t1>* *<var1>* . . .

where the time values *<t0>*, *<t1>*, . . . are of data type *t-real*, and the entries *<var0>*, *<var1>*, . . . are the corresponding values of the primary variable that is being specified. (See Section 6 for an important note.) Primary variable values are calculated from the table using linear interpolation.

factor

This data unit is optional and is used to modify the component mass fluxes by a factor that can depend on time and that is set by a table. One use of this option is to turn off a component flux coming out of or into a boundary element at certain time intervals or at all times. Note that several different components can be specified, each having its own time-dependent factor. Tables for all of the components must be present (the components are model-specific). Leaving the *factor* data unit out is equivalent to specifying a modification factor of 1.0 for all time.

phasefactor

This data unit is *optional* and is used to modify the phase fluxes by a factor that can depend on time and that is set by a table.

<comp-name> *word*

Name of the component flux that will have its flux modified (model-specific)

<phase-name> *word*

Name of the phase that will have its flux modified (model-specific)

<comp-factor-table> *table*

Table of modification factors for component fluxes

It is of the form,

$\langle t0 \rangle \ \langle fac0 \rangle \ \langle t1 \rangle \ \langle fac1 \rangle \ . \ . \ .$

where the time values $\langle t0 \rangle, \langle t1 \rangle, \dots$ are of data type *t-real*, and the entries $\langle fac0 \rangle, \langle fac1 \rangle, \dots$ are the corresponding modification factors. (See Section 6 for an important note.) Modification factors are calculated from the table using linear interpolation.

<phase-factor-table> *table*

Table of modification factors multiplying all component fluxes in the corresponding phase given by *<phase-name>*; format is analogous to *<comp-factor-table>*

(clamped)

Keeps the primary variables for these elements, as set in `state`, fixed in time

NOTES

The data: (`factor . . .`) is optional; if not present, factors will be unity.

Currently, the model may, in some cases, choose time steps so large that it overshoots sharp changes in a time table. This may be a serious problem if, for example, one wishes to model a flux that is turned off suddenly. A solution is to use the `forcetimes` command in `output`; this forces the model to hit specified times; in this case, the times at which the boundary condition changes suddenly.

The model will abort unless the last time value in `table` is greater than or equal to the ending time of the simulation as set by `tstop`.

5.9 Other Options

NAME

element-prefix-delimiter, element-indices-separator

Set format of element names

SYNOPSIS

```
[ (element-prefix-delimiter "<prefix-sep>" ) ]  
[ (element-indices-separator "<ind-sep>" ) ]
```

DESCRIPTION

Changes the format of element names created by `genmsh`.

NOTES

The `genmsh` command names the element according to the general format

<elem-prefix>#<i>:<j>:<k>

where *<I>*, *<j>*, *<k>* denote the *i*, *j*, and *k* indices of the element, and *<elem-prefix>* is set by the `mat` command inside `genmsh`. These commands allow the user to change the separators “#” and “:” by the single character in the strings “*<prefix-sep>*” and “*<ind-sep>*”, respectively.

6. Running Flow and Transport Sequentially

In some modules, such as the `us1c` module, the flow of the fluid phases is first calculated at the beginning of each step, and then the transport of contaminants is performed using the resulting flow field. (Other modules, such as `usnt`, solve for flow and transport simultaneously.) When the flow and transport models are solved separately, each model has its own set of input data units in the same input data file. Data units that are common to both models can be placed in a common data unit called `(common . . .)`, which holds data units used by both the flow and transport models. If a data unit appears in both the `common` data unit and the particular unit belonging to the model, the one in `common` takes precedence.

General Form for Running Flow and Transport Sequentially

```
(common
  (title . . .)
  (outputfile . . .) || {(output-prefix . . .)(output-ext . . .)}
  (meshfile . . .) || (genmsh . . .)
  (time . . .)
  (tstop . . .)
)
(<flow-model-name>
  (dt . . .)
  (dtmax . . .)
  (stepmax . . .)
  (state . . .)
  (rocktab . . .)
  [ (output . . .) ]
  [ (srctab . . .) ]
  [ (bctab . . .) ]
)
(<transport-model-name>
  (dt . . .)
  (dtmax . . .)
  (stepmax . . .)
  (state . . .)
  (rocktab . . .)
  [ (output . . .) ]
  [ (srctab . . .) ]
  [ (bctab . . .) ]
  [ (phaseprop . . .) ]
  [ (compprop . . .) ]
)
```

7. References

Bear, J., and Y. Bachmat. *Introduction to Modeling of Transport Phenomena in Porous Media*, Kluwer Acad. Publishers. 1991.

Behie, A., and P.K. Vinsome. "Block iterative methods for fully implicit reservoir simulation," *Soc. Petroleum Engineers*, paper 9303. 1980.

Edwards, A.L. "TRUMP: a computer program for transient and steady-state temperature distributions in multidimensional systems." Springfield, VA: National Tech. Information Service. 1972.

Grabowski, J.W., P.K. Vinsome, R.C. Lin, A. Behie, and B. Rubin. "A fully implicit, general-purpose, finite-difference thermal model for in situ combustion and steam," *Soc. Petroleum Engineers*, paper 8396. 1979.

Lee, K., A. Kulshrestha, and J. Nitao. "Interim Report on Verification and Benchmark Testing of the NUFT Computer Code." Livermore, CA: Lawrence Livermore National Laboratory. UCRL-ID-113521. 1993.

Narasimhan, T.N., and P.A. Witherspoon. "An integrated finite difference method for analyzing fluid flow in porous media," *Water Resour. Res.* **14** 255-261. 1978.

Richtmyer, R.D., and K.W. Morto. *Difference Methods for Initial-Value Problems*. New York, NY: Interscience Pub. 1967.

Smolarkiewicz, P.K. "A fully multidimensional positive definite advection transport algorithm with small implicit diffusion," *J. Computational Physics*. **54**, 325. 1984.

Vinsome, P.K.W. "Orthomin, an iterative method for solving sparse sets of simultaneous linear equations," *Soc. Petroleum Engineers*, paper 5729. 1976.

Appendix A. Format of the Mesh File

The mesh file is free format and can be either of two forms.

First Form

The first form uses element names to specify the connections between two elements:

```

$elc
% <el-name> <rock-type> <volume> <xc> <yc> <zc>
. . .
% <el-name> <rock-type> <volume> <xc> <yc> <zc>
$end
$con
% <el-name0> <el-name1> <isot> <L0> <L1> <A> <beta>
. . .
% <el-name0> <el-name1> <isot> <L0> <L1> <A> <beta>
$end

```

Second Form

The second form uses element numbers to specify connections:

```

$eleme
% <el-num> <el-name> <rock-type> <volume>
. .
% <el-num> <el-name> <rock-type> <volume>
$end
$conne
% <mp0> <mp1> <isot> <L0> <L1> <A> <beta>
. . .
% <mp0> <mp1> <isot> <L0> <L1> <A> <beta>
$end

```

<el-name>	element name (<i>word</i>)
<rock-type>	rock type of element (<i>word</i>)
<volume>	volume of element (<i>real</i>)
<xc>, <yc>, <zc>	coordinates of element centroid

<i><el-name0></i> ,	specifies a connection with positive flow going from element <i><el-name0></i> to <i><el-name1></i> (<i>word</i>)
<i><el-name1></i>	value of 0, 1, or 2 referring to which conductivity <i>K0</i> , <i>K1</i> , or <i>K2</i> in <i>(rocktab . . .</i> is used for this connection (<i>integer</i>)
<i><isot></i>	lengths of the parts of the connection that lie in the first and second elements that define the connection, respectively (<i>real</i>)

where

<i><A></i>	flow area of a connection (<i>real</i>)
<i><beta></i>	cosine of the angle of vertical inclination of the connection (<i>real</i>)
<i><el-num></i>	the element number starting from 0; must be consecutive (<i>integer</i>) in <i>\$eleme</i> ; used primarily for convenience by the user in associating connections with the elements they connect when the <i>\$eleme</i> and <i>\$conne</i> options are used
<i><mp0></i> , <i><mp1></i>	specifies a flow connection with positive flow going from <i><mp0></i> to <i><mp1></i> (<i>integer</i>)

The advantage of the first format is that connections are easily identified by looking at the element names that they connect, especially if the names are chosen in a convenient manner. The disadvantage is that more initial set-up time is needed by the model to associate the element names with the element numbers that will be used during the simulation. For use with more than 4000 elements, the second format is recommended.

Appendix B. Numerical Algorithms Used

B.1 Numerical Discretization

NUFT solves balance equations (see, for example, Bear and Bachmat [1991]), which are partial differential equations whose specific form depends on the particular NUFT module being used. The balance equations are discretized in space using the integrated, finite-difference method (see Edwards [1972], Narasimhan and Witherspoon [1978]). Spatial discretization results in a system of ordinary differential equations of the form

$$\frac{\partial \mathbf{u}}{\partial t} = \mathbf{f}(\mathbf{u}, t) \quad (1)$$

where \mathbf{u} is the solution vector and \mathbf{f} is a vector-valued function.

A first-order, implicit-in-time scheme (Richtmyer and Morton, 1967) is used to discretize the balance equations for modules with combined flow and contaminant transport. For modules where flow and transport are solved sequentially, the flow equations are solved using the implicit-in-time method, and the transport equations can be solved either implicitly or explicitly in time depending on the user's choice.

When Eq. (1) is solved using the implicit-in-time method, the resulting system of equations that must be solved at each time step is nonlinear if \mathbf{f} is nonlinear.

$$\mathbf{F}(\mathbf{u}^{(n+1)}) \equiv \mathbf{u}^{(n+1)} - \mathbf{u}^{(n)} - \Delta t^{(n)} \mathbf{f}(\mathbf{u}^{(n+1)}, t^{(n+1)}) = 0 \quad (2)$$

where superscripts represent the time level.

B.2 Newton-Raphson Iteration

The nonlinear system of Eq. (2) is of the form

$$\mathbf{F}(\mathbf{u}) = 0 \quad (3)$$

The Newton-Raphson iterative method is used to solve the equations

$$\mathbf{J}(\mathbf{u}_n)(\mathbf{u}_{n+1} - \mathbf{u}_n) = -\mathbf{F}(\mathbf{u}_n) \quad (4)$$

where the matrix

$$\mathbf{J}(\mathbf{u}_n) \equiv \left[\frac{\partial F_i}{\partial u_j} \right] \quad (5)$$

is the Jacobian of the function \mathbf{F} evaluated at the n -th iterate, \mathbf{U}_n . Note that Eq. (4) can be viewed as a system of linear equations:

$$\mathbf{Ax} = \mathbf{b} \quad (6)$$

where

$$\mathbf{A} = \mathbf{J}(\mathbf{u}_n) \quad (7)$$

$$\mathbf{x} = \mathbf{u}_{n+1} - \mathbf{u}_n \quad (8)$$

$$\mathbf{b} = -\mathbf{F}(\mathbf{u}_n) \quad (9)$$

B.3 Solution of System of Linear Equations

The code currently has two different options for solving the system of linear equations: banded gaussian elimination and preconditioned conjugate gradient methods.

B.3.1 Banded Gaussian Elimination

First, for convenience, Eq. (4) is rewritten in the matrix form

$$\mathbf{Ax} = \mathbf{b} \quad (10)$$

where $\mathbf{A} = [\mathbf{A}_{i,j}]$ is an $n \times n$ matrix. It is well known that, in many problems, the matrix \mathbf{A} is *banded* (i.e., $\mathbf{A}_{i,j} = 0$ for $|i - j| > n_b$ where n_b is the matrix half-bandwidth). If $n_b \ll n$, it is advantageous to perform the gaussian elimination only on the elements within the band. This band becomes more or less filled with non-zero elements as the elimination proceeds.

Problems where \mathbf{A} is banded include the case where the problem domain is discretized by a rectangular mesh and the elements are ordered either rowwise or columnwise.

B.3.2 Orthomin Preconditioned Conjugate Gradient

For the description of the orthomin algorithm and the preconditioning options, see Vinsome (1976) and Behie et al. (1980).

B.4 Automatic Time-Stepping

The time-stepping algorithm seeks to control the maximum change in the solution vector from one time step to the next. This control can be accomplished by estimating the time step for the solution to change by a specified amount based on the solution change that occurred in the previous two time steps. During an iteration, if the change in solution from the previous time step is too large, a re-estimation is performed, and the time step is started over. This method also controls, to some extent, the time-discretization error.

The method is now described in more detail: Let u_i be the i -th component of the solution vector at the current iteration of the Newton-Raphson method, and δu_i be equal to u_i minus the i -th component of the solution vector at the end of the previous time step. Now, let $(\delta u_i)_{max}$ be the maximum allowed change in the solution. The new reference time step is defined as δt by

$$\delta t = \frac{(1+w)(\delta u_i)_{max}}{w(\delta u_i)_{max} + \delta u_i} \quad (11)$$

Here, w is a “damping” factor (Grabowski et al., 1979) that is chosen between 0 to 1 to prevent the time step from changing too rapidly. A value of 0.8 is used. The value of δt is adjusted to stay within 0.5 to 4.0 times the time-step size taken in the previous time step. If the reference time step is less than the current time step because the solution changed too much, the current time step is replaced by 0.8 times the reference, and the Newton-Raphson is restarted. The factor 0.8 is there to allow for some margin to prevent an excess of restarts.

If the equations are solved using an explicit-in-time method, the time step is restricted so that the Courant-Friedrichs-Levy criteria (Richtmyer and Morton, 1967) is satisfied.